NASA-CR-191690

NUMERICAL INVESTIGATIONS IN THREE-DIMENSIONAL INTERNAL FLOWS

SEMI-ANNUAL STATUS REPORT

1 JULY THROUGH 31 DECEMBER 1992

Prepared for:

NASA-AMES RESEARCH CENTER

MOFFETT FIELD, CA 94035

UNDER NASA GRANT

NCC 2-507

By:

WILLIAM C. ROSE

ENGINEERING RESEARCH AND DEVELOPMENT CENTER

UNIVERSITY OF NEVADA, RENO

RENO, NV 89557

I. BACKGROUND

NASA has an ongoing interest in supersonic and hypersonic inlet flow field research. Their research efforts are intended to complement prospective aerospace vehicles, such as the High-Speed Civilian Transport (HSCT) and the National Aerospace Plane (NASP), as well as other variants of these vehicles intended for use with air-breathing propulsion systems. Computational Fluid Dynamics (CFD) is expected to play a large part in the design and analysis of such aircraft because experimental facilities are limited. The purpose of this Grant is to apply, evaluate and validate CFD tools for use in high-speed inlet flow fields.

In previous efforts under the current Grant, a two-dimensional full Navier-Stokes (FNS) code (SCRAM2D) was used in a design process that involved parametric modifications of the inlet geometry to arrive at what appeared to be an optimum inlet flow field that produced a uniform flow at the exit in a very short distance. In these previous studies, the technologies for determining the contours with a "man-in-the-loop" approach for both the ramp and cowl of the inlet were demonstrated and nearly shock-free exiting flow fields were shown to be obtainable. The resulting two-dimensional compression contours were then used with swept sidewalls to form a three-dimensional inlet. Then the three-dimensional Navier-Stokes code (SCRAM3D) was used to investigate the inlet's three-dimensional flow.

Also in previous efforts under the current Grant, 2D and 3D space-marched Navier-Stokes codes were applied to inlet flow field analysis. It was shown that by using the STUFF code, a space-marched, thin-layer Navier-Stokes code developed and used by Greg Molvik of the MCAT Institute at Moffett Field, California, considerable time reductions could be obtained by solving the flow field that behaves parabolically and then matching the output of that code to the input of an FNS code such as SCRAM2D or SCRAM3D (or the time-marched version of STUFF called TUFF). In a previous status report, the validity of this process and the accuracy of using the STUFF code was demonstrated by direct comparison with experimental data obtained from the NASA-Lewis Mach 5 inlet study.

Most efforts conducted to date have examined the flow field characteristics and performance of isolated inlet systems. In reality, of course, these inlet systems are installed on aircraft at various locations. From an inlet standpoint, the primary effect of the installation of the inlet on an aircraft is to modify the incoming boundary layer thickness relative to the cowl height. In the present reporting period, the hybrid method of using the STUFF and SCRAM2D codes was applied to a forebody that is representative of a waverider configuration being considered for a long-range cruise application at a design Mach number of 5. This report describes the results of the investigation into the potential application of the NASA-Lewis/NASP Mach 5 inlet contours to this waverider aircraft.

II. INTRODUCTION

In the present reporting period, the 2D versions of STUFF and SCRAM2D have been applied to solve the forebody flow and resulting inlet flow field for the hypothetical aircraft shown with the surface grid in Figure 1. Although the flow is expected to be three-dimensional, particularly near the nose of the vehicle, it was felt that a useful approximate solution could be obtained quickly using a 2D version of STUFF to solve the forebody flow field and initial external ramp. Since the actual aircraft might be expected to have multiple propulsion cycles used throughout the range of operating Mach numbers from 0 to 5, other modified flow paths were considered in the present study. Two cases were examined. The first case was the Mach 5 on-design configuration (with its single flow path) and the second was an off-design case at Mach 3, for which a dual flow path was considered.

III. RESULTS AND DISCUSSION

The geometry shown in Figure 1 was used to obtain the contours along the centerline of the aircraft. These contours were then used to obtain the flow field solution for the Mach 5, on-design condition with a hypothetical operating altitude of 100,000 feet above sea level. This waverider aircraft has a 4.1 degree deflection ramp ahead of the installation of the Mach 5 NASA-Lewis/NASP inlet. This 4.1 degree ramp is intended to reduce the vehicle flow field Mach number to a value of 4.1, which was the design Mach number entering the inlet on the Lockheed Mach 5 penetrator aircraft (the basis of the original Mach 5 inlet design). A solution obtained for the entire forebody flow field and inlet is shown in terms of the Mach contours for the design condition in Figure 2. This shows the bow shock from the vehicle, the forebody boundary layer, the single shock wave from the additional 4.1 degree ramp, and, finally, the three shock waves from the Mach 5 inlet which coalesce near the cowl lip. A detailed view of the on-design solution from just upstream of the cowl lip is shown in Figure 3. The Mach number contours indicate the single-shock wave from the 4.1 degree additional deflection, plus the combination of the three shock waves from the three inlet ramps. In addition, the boundary layer accumulated from the forebody is clearly seen. This boundary layer is approximately 20% thicker than the corresponding boundary layer examined in the NASA-Lewis Mach 5 inlet wind tunnel test. Because of the increased boundary layer thickness, the cowl shock wave is felt on the ramp surface further upstream than in the design or the wind tunnel test. Boundary layer bleed, similar to that required in the wind tunnel to maintain the inlet operating supercritically, is required. The cowl shock wave pressure rise being significantly further upstream than in the wind tunnel test implies that a redesign of the inlet contour on the ramp side will be required in order to maintain an operating margin. The STUFF code was used to a streamwise location just downstream of the leading edge of the cowl. The remainder of the solution is from the SCRAM2D code.

The currently hypothesized aircraft described with the aid of Figure 1 has a dual propulsion system. A turbojet is envisioned to operate between take off and approximately Mach 3, while a ramjet system is intended to operate from approximately Mach 3 to the design condition. Near the tradeoff Mach number of 3, both paths must be successfully operating and providing combustible air to the turbojet and the ramjet systems. This dual path presents a challenge to inlet design that has not been previously investigated using any modern CFD technique. In the present study, a hypothetical bifurcation of the Mach 5 inlet contours was investigated to demonstrate the applicability of the hybrid STUFF and SCRAM2D approach to this inlet analysis. The entire vehicle line and the dual path inlet solution are shown in Figure 4. This case was calculated for a Mach 3 condition with an altitude of 70,000 feet above sea level. The Mach 3 design contours were used for the ramjet portion of the flow, while the turbojet portion is a new hypothetical contour derived in this study from the original Mach 5 inlet by pivoting a portion of the ramp surface located just above the cowl lip into the flow to provide a flow path for the turbojet system. A solution obtained for this bifurcated flow path is shown in detail in Figure 5. Here, the Mach number contours are shown for a dual-path arrangement with a steady-state subsonic flow achieved in both paths. Bleed is required in both paths to stabilize the terminal shock waves that produce subsonic flow. This solution demonstrates the power of the technique of using the hybrid STUFF and SCRAM2D codes.

In a later portion of the study using the dual-flow-path arrangement, the question arose as to the nature of the flow field that would be produced if, for example, the turbojet inlet were to unstart. A time-dependent solution was obtained by back-pressuring the engine face station of the turbojet path in order to expel the terminal shock wave system. Selected flow field states in the time-dependent solution are shown in Figure 6. The solution indicates that an unstart in this hypothetical, dual-path arrangement of the turbojet would also lead to an unstart of the ramjet system without some sort of active bleed-control system to increase the stability margin of the lower ramjet path. A video of this unstart process was made during the course of the study and is available.

IV. CONCLUSIONS

An investigation was carried out into the feasibility of using the hybrid space-marched/time-marched Navier-Stokes codes, STUFF AND SCRAM2D, developed in the last reporting period, as applied to a hypothetical aircraft design. The use of the two codes allowed significantly reduced computational times while maintaining sufficient accuracy to resolve the details of the inlet flow path. The codes were applied to solve the flow over a hypothetical aircraft at its design Mach number of 5 and an off-design Mach number of 3. A single path ramjet flow was assumed for the design case, while a dual turbojet and ramjet path was assumed for the off-design case. A time-dependent solution was obtained in which back pressure at the turbojet engine face was increased enough to cause an inlet unstart that ultimately led to an unstart in the ramjet path of the dual-flow inlet. Future efforts will concentrate on applying the three-dimensional versions of both of these codes to the full waverider configuration and its inlet sidewalls.

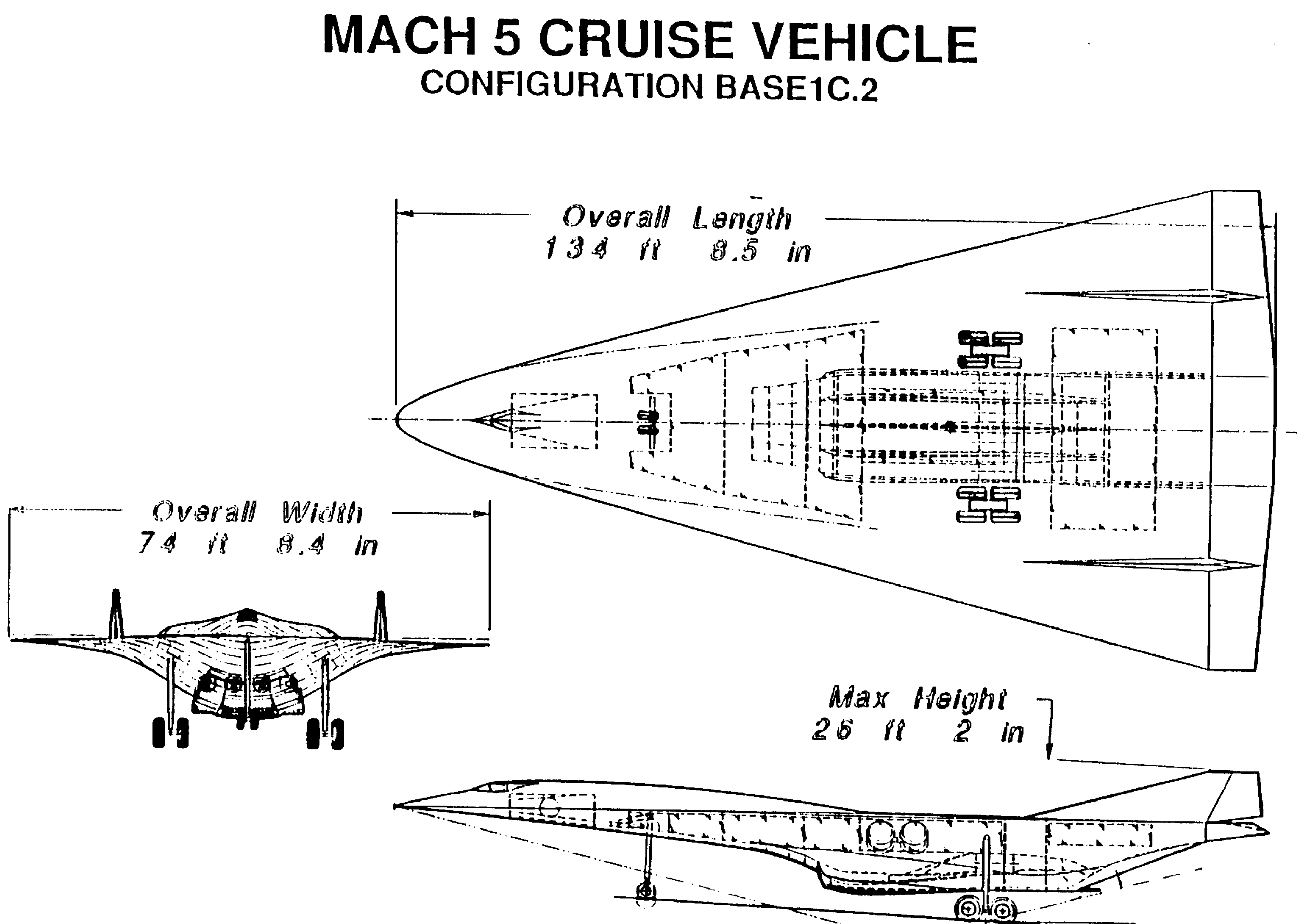


FIGURE 1 Configuration for a Mach 5 waverider cruise vehicle.

a) 3 views

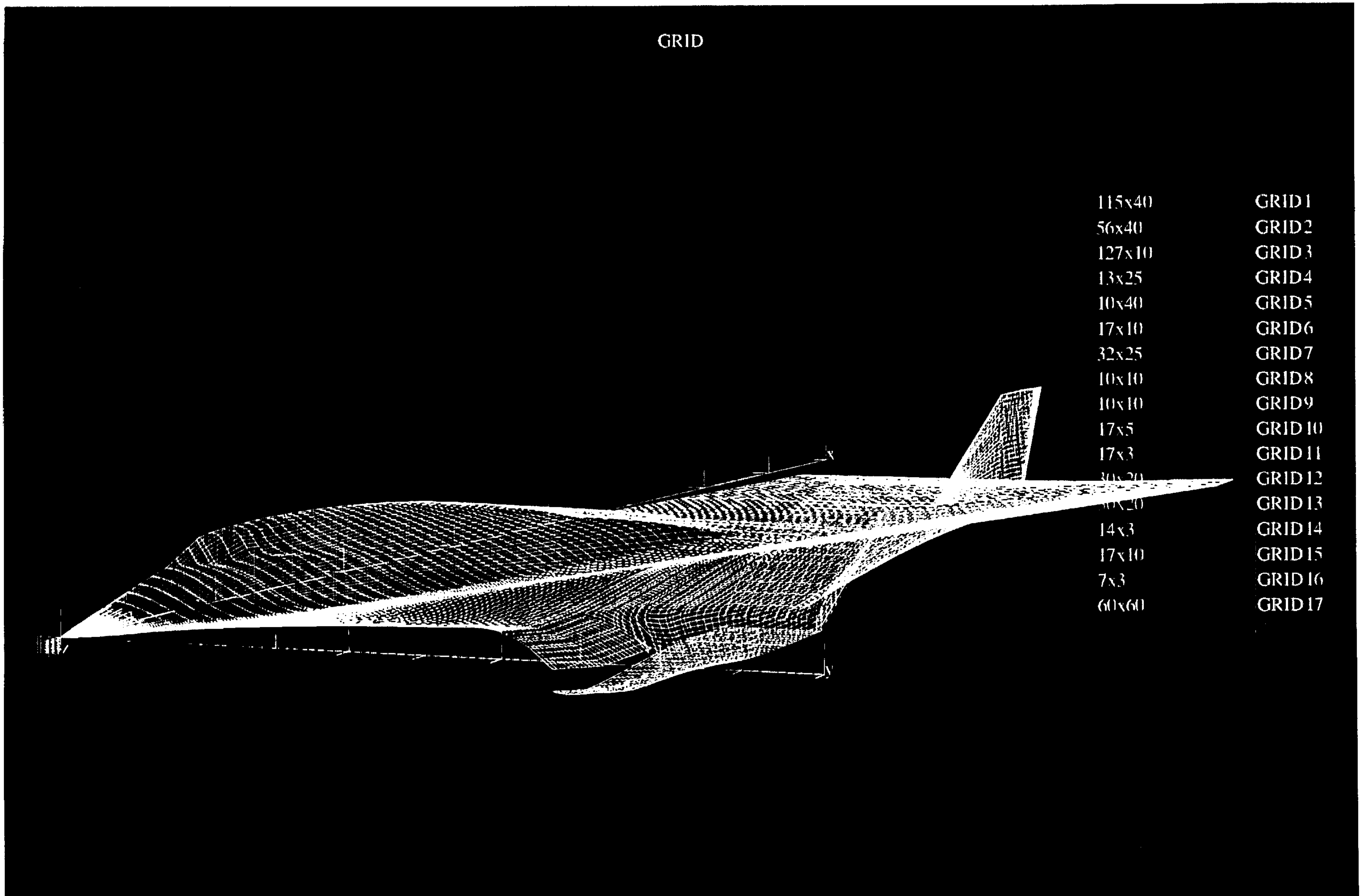


FIGURE 1 Concluded.

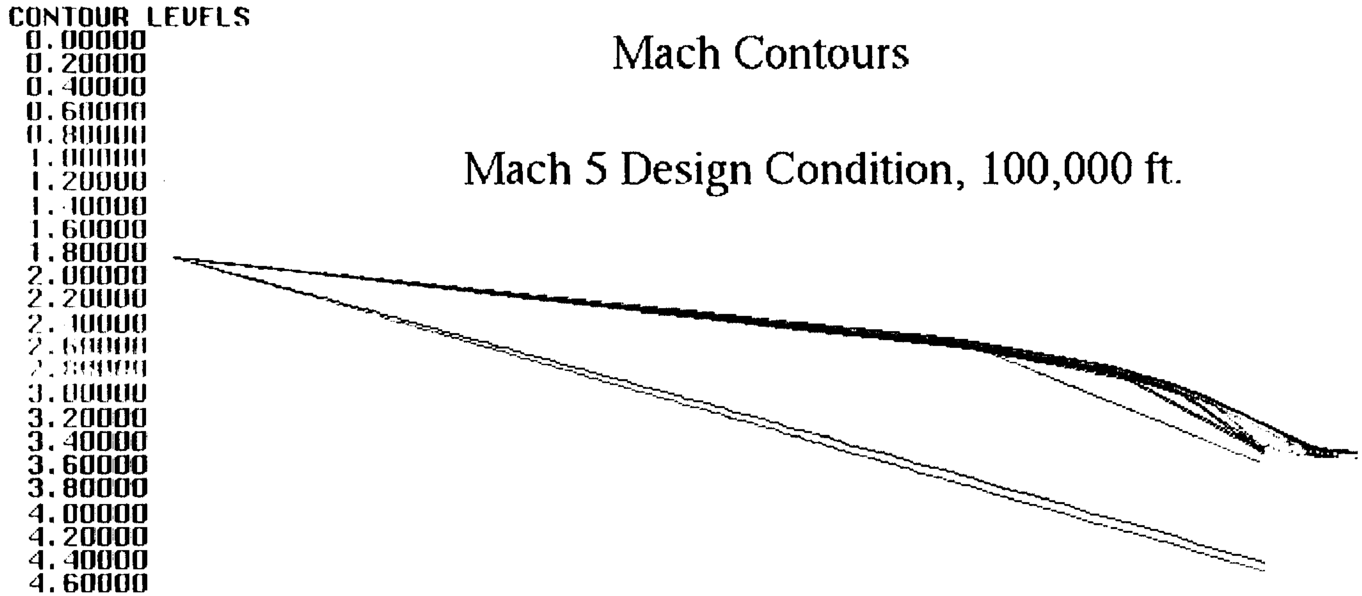
b) Perspective with surface grids.

Mach 5 Waverider

Long Range Cruise Study

Mach Contours

Mach 5 Design Condition, 100,000 ft.



Cowl lip station boundary layer thickness is larger percentage of lip height than wind tunnel case.

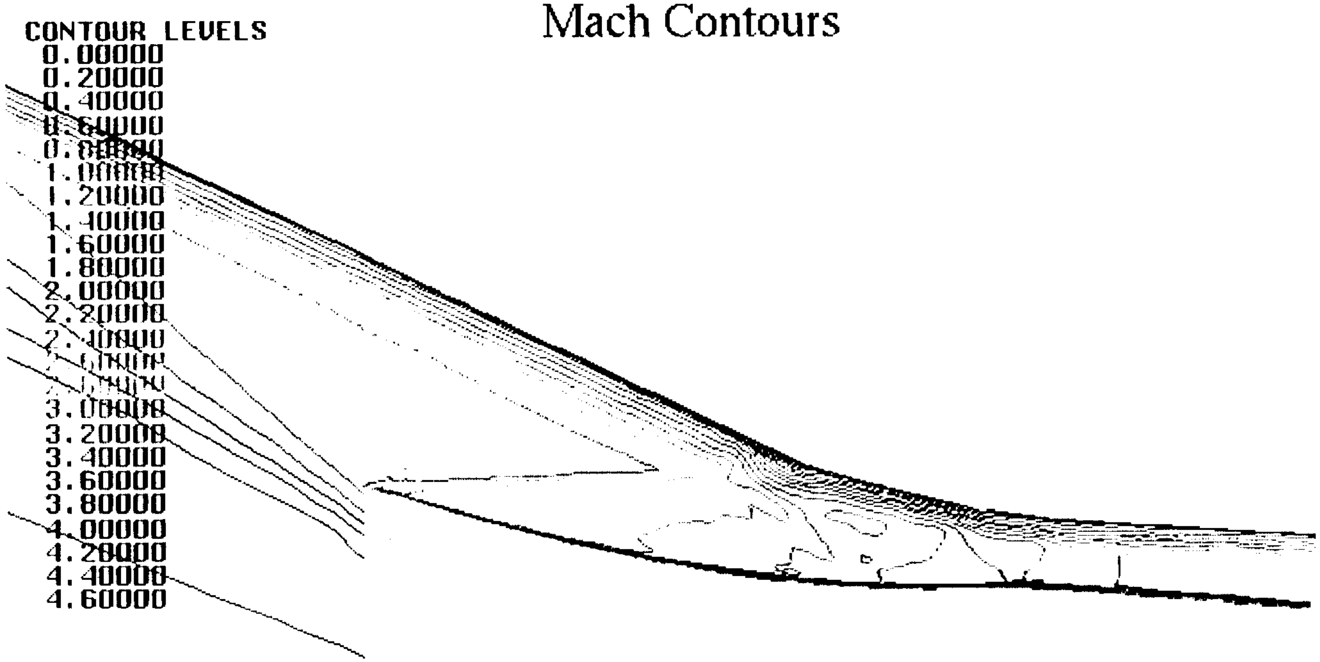
More bleed may be required.

FIGURE 2 **Mach contours at Mach 5 from 2D solution for center plane of vehicle including forebody and inlet.**

Mach 5 Waverider

Long Range Cruise Study

Mach Contours



Mach 5 Design Condition, 100,000 ft.

Ramjet path only.

Bleed similar to wind tunnel locations required.

FIGURE 3 Detail of Mach 5 inlet flow.

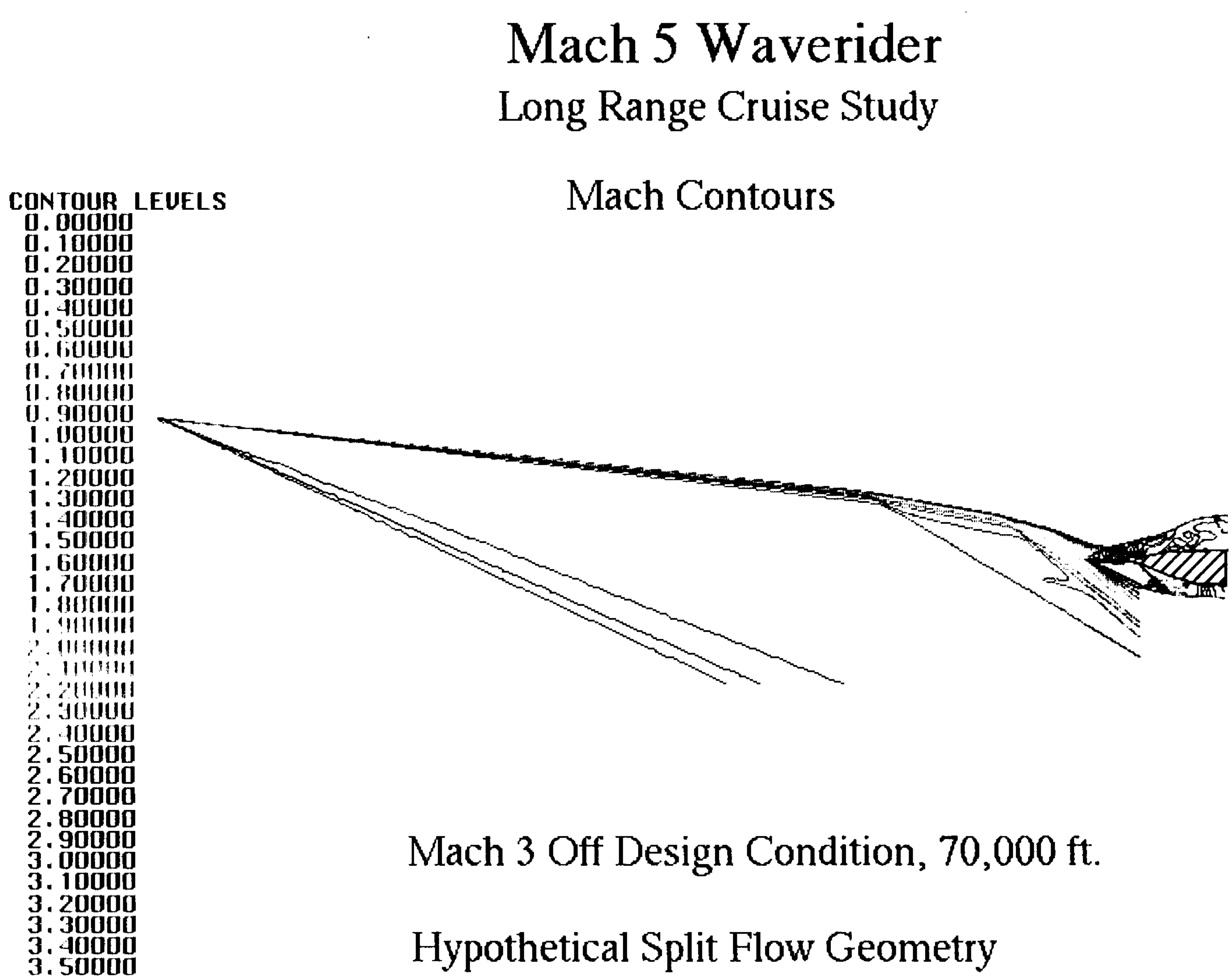


FIGURE 4 **Mach contours at Mach 3 from 2D solution for center plane of vehicle including forebody and dual-path inlet.**

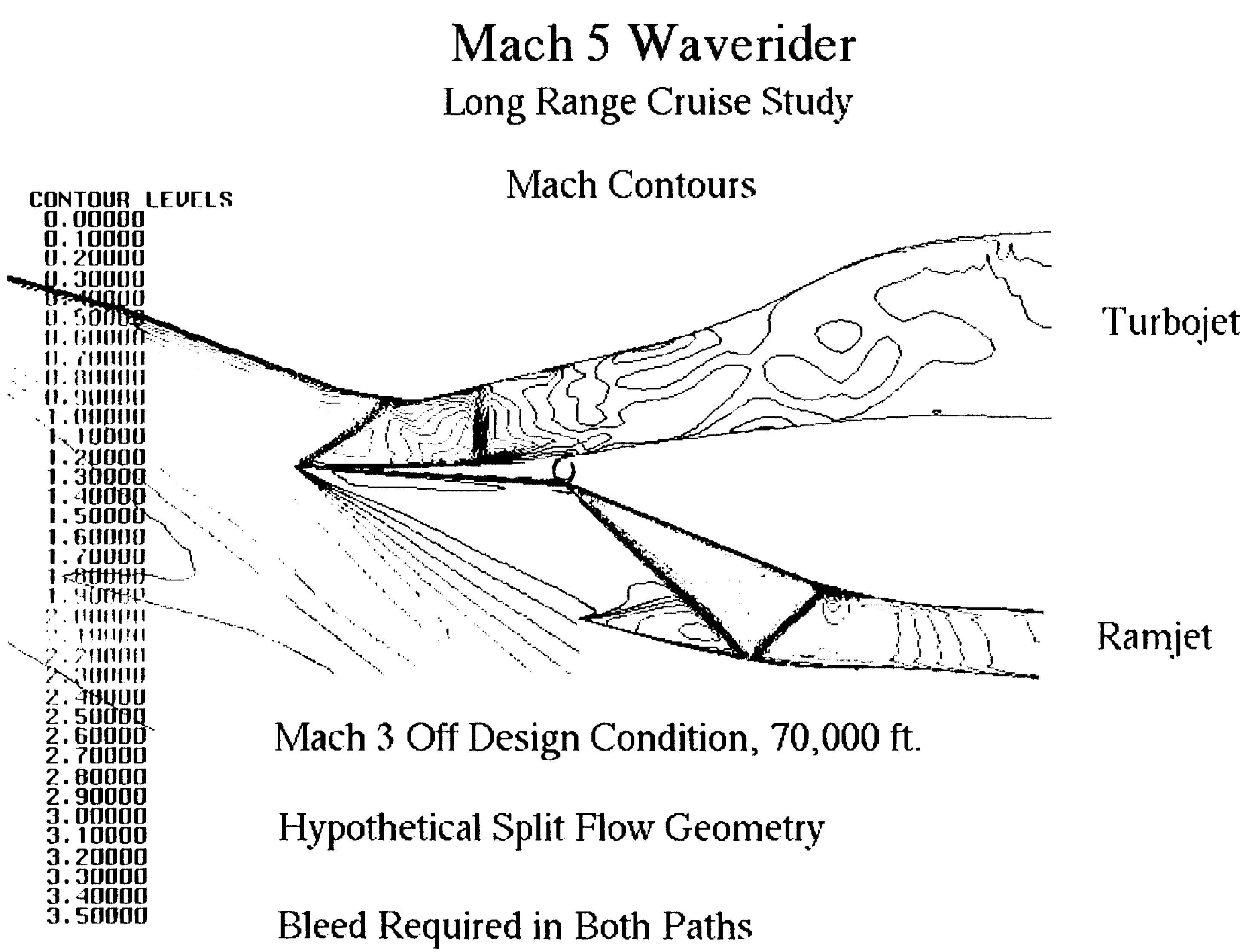


FIGURE 5 **Detail of Mach 3 dual-path inlet flow.**

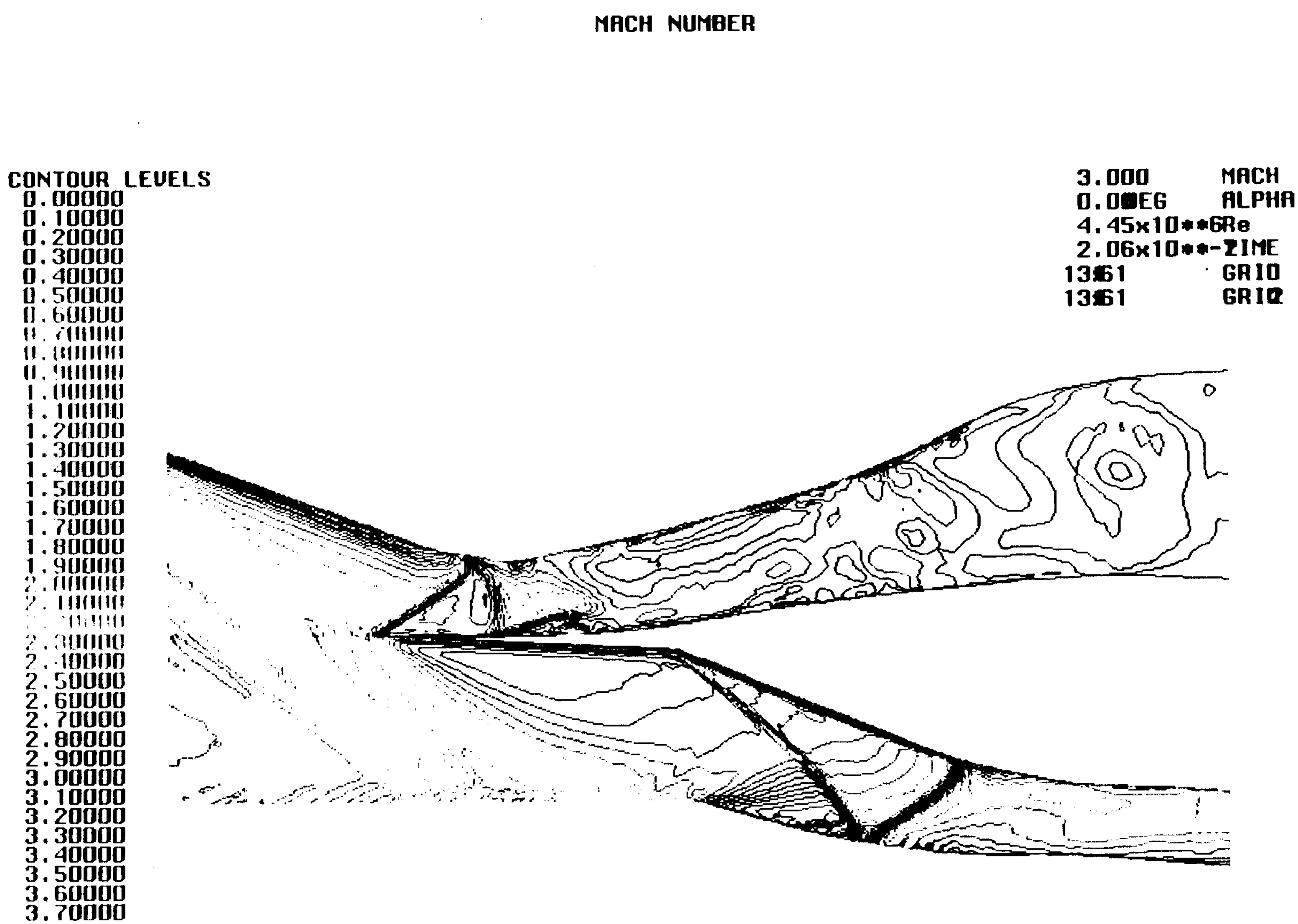


FIGURE 6 Time evolution of an unstart initiated in the turbojet (upper) path.

a) Time 1

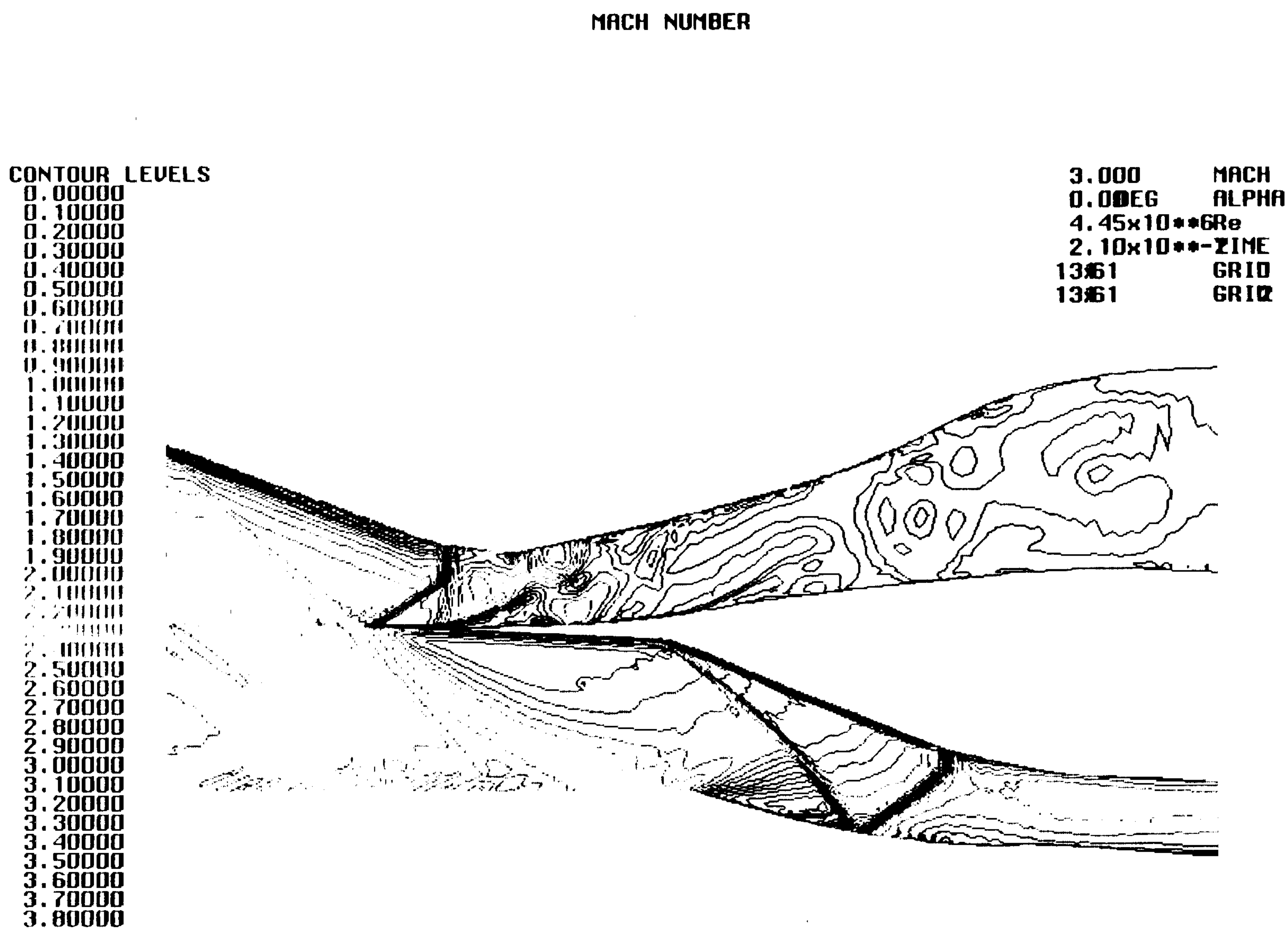


FIGURE 6 Continued.

b) Time 2

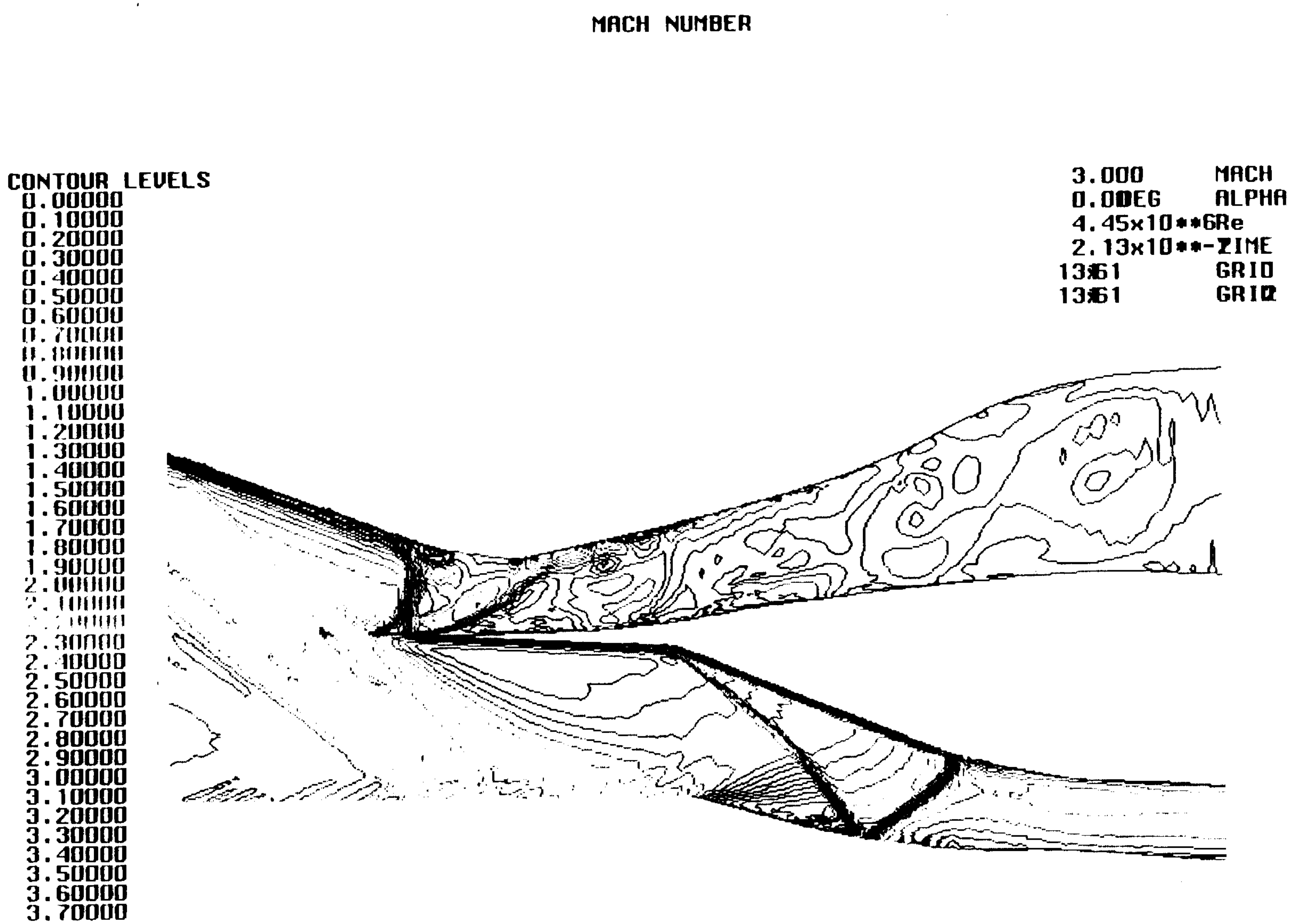


FIGURE 6 Continued.

c) Time 3

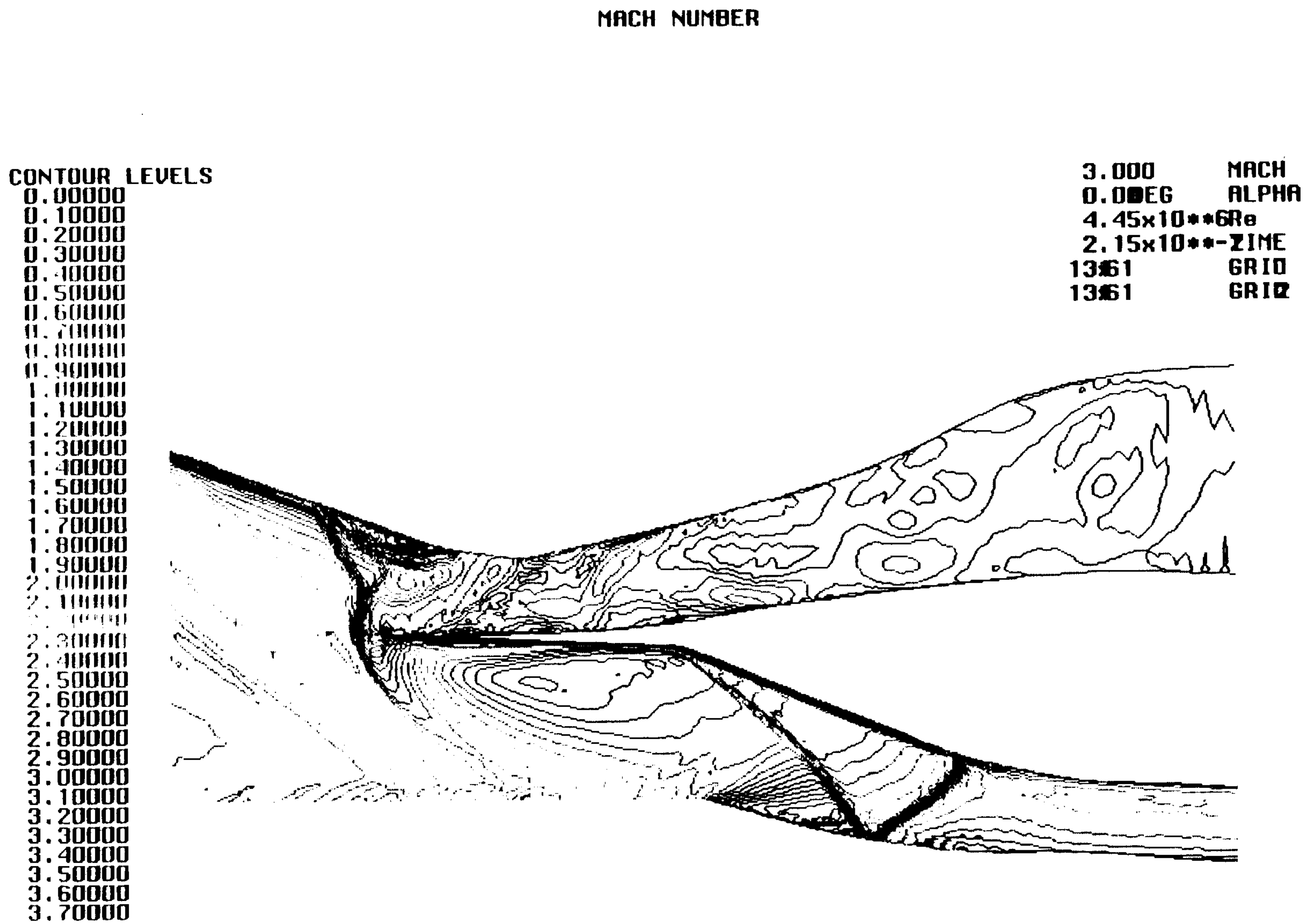


FIGURE 6 **Continued.**

d) **Time 4**

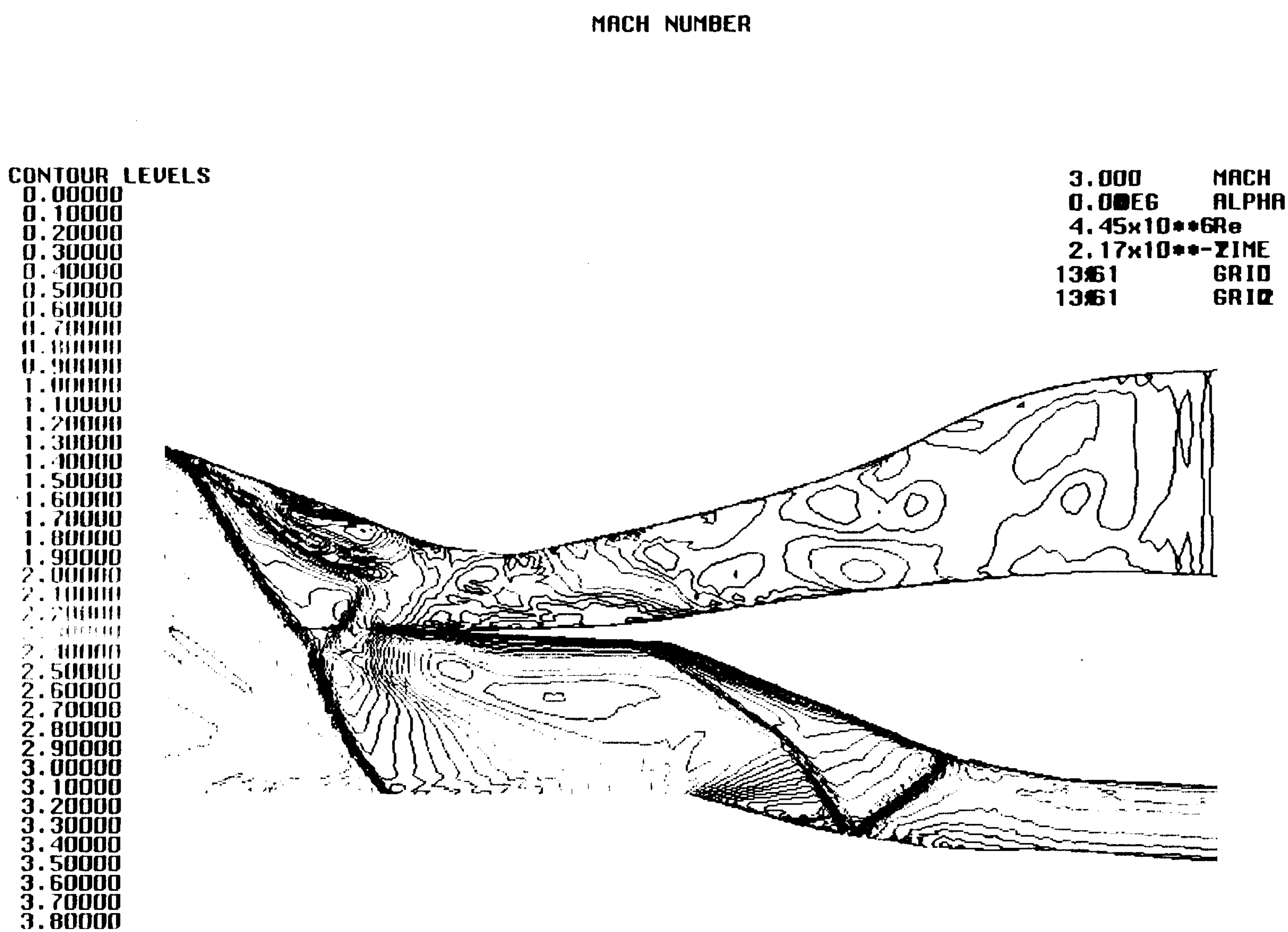


FIGURE 6 **Continued.**

e) **Time 5**

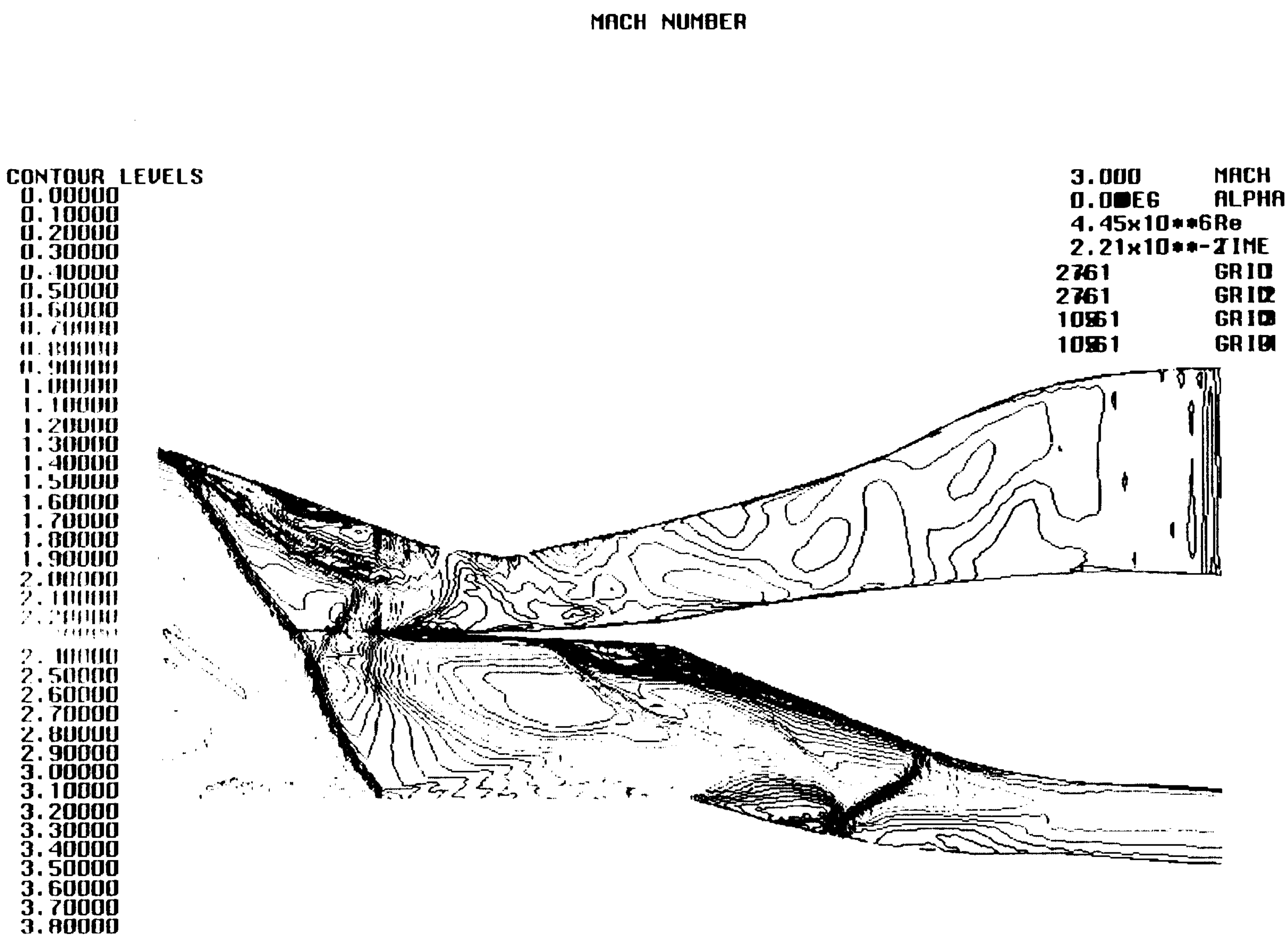


FIGURE 6 Continued.

f) Time 6

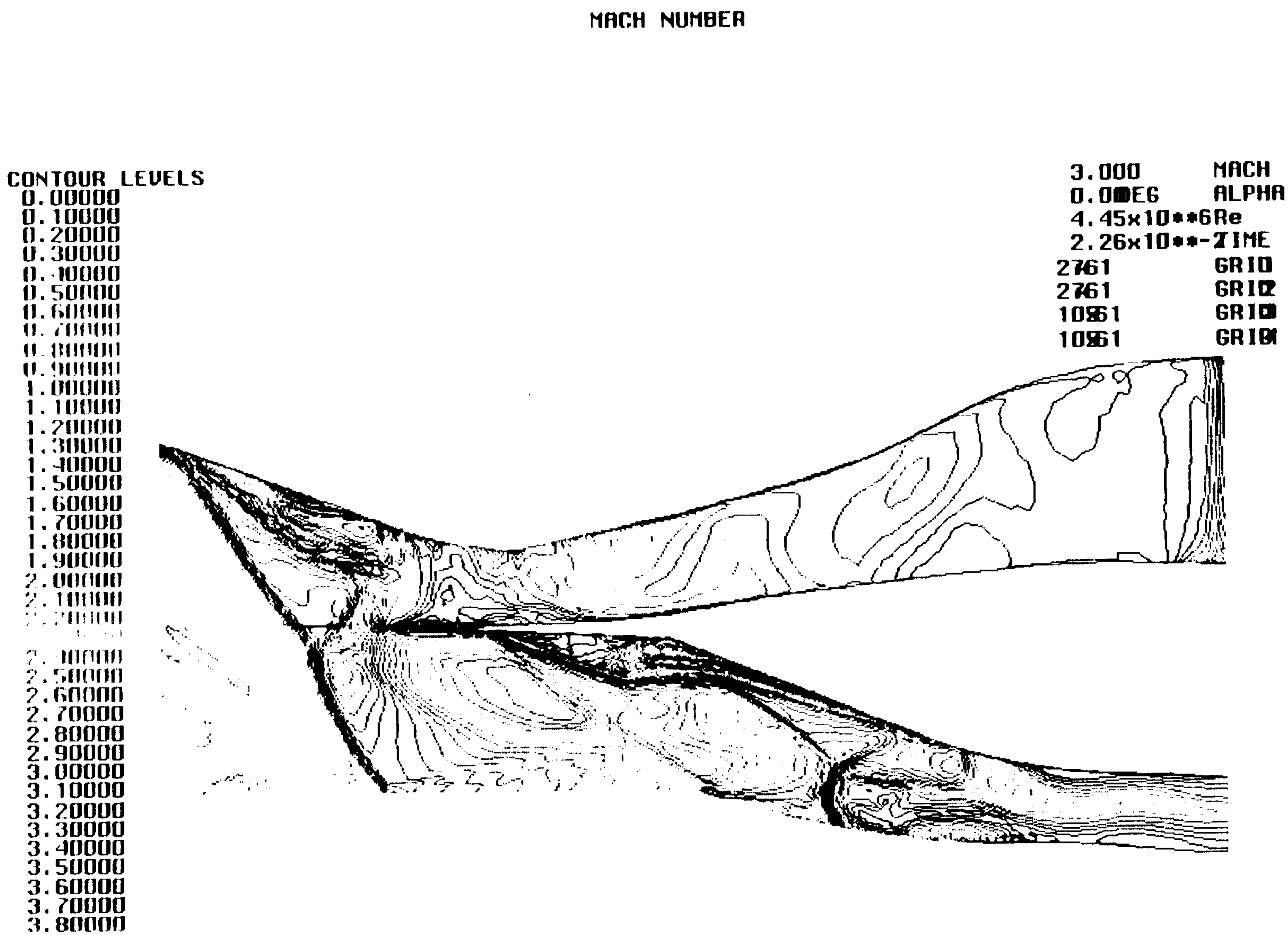


FIGURE 6 Concluded.

g) Time 7